Numerical simulation of supersonic jet noise using open source software [⋆]

Andrey Epikhin[1,2,3][0000−0002−6249−4283] and Ivan But[1,2][0000−0001−5915−0941]

[1] Ivannikov Institute for System Programming of the RAS, 109004, Russia
[2] Keldysh Institute of Applied Mathematics of the RAS, 125047, Russia
[3] Bauman Moscow State Technical University, 105005, Russia

Abstract. The paper is devoted to the study of various numerical algorithms for calculating the flow and acoustics characteristics of supersonic jets implemented in open source software. The ideally expanded supersonic jet with parameters $M = 2.1$, $Re = 70000$ is considered. A comparison of various approaches implemented in the OpenFOAM and block-structured adaptive mesh refinement framework of AMReX is conducted. Numerical algorithms for compressible gas flow implemented in pimpleCentralFoam, QGDFoam and CNS solvers are considered. Acoustic noise are calculated using the Ffowcs Williams and Hawkings analogy implemented in the libAcoustics library. Cross-validation comparison of the flow fields and acoustic characteristics is carried out.

Keywords: Aeroacoustics · Noise · Jet · Compressible flow · Quasi-gas dynamic equations · OpenFOAM · AMReX.

1 Introduction

The relevance of the research topic is determined by the prevalence of jet streams in nature and technology. Laminar jets are quite rare in nature, therefore, in the future, more attention was paid to both theoretical and experimental works on turbulent jets [1].

There are two main approaches for studying jet flows: full-scale experiments [2, 3] and numerical simulations [4, 5]. Numerical simulations are more cost-effective, making them more popular. There are various methods used in numerical simulations, such as high-order accuracy methods and the Galerkin method [6], Godunov-type approximation methods (Kurganov-Tadmor, Rusanov, HLLC, etc.), hybrid approach [8], and algorithms based on regularized (quasi-gasdynamic) equations of gas dynamics [9]. Each method has its own advantages and limitations, and choosing the right method depends on the specific goals of the study. For instance, Godunov-type approximation methods are limited in their applicability to Mach numbers larger 1, making it difficult to use them for subsonic

---

[⋆] Supported by Moscow Center of Fundamental and Applied Mathematics, greement with the Ministry of Science and Higher Education of the Russian Federation, No. 075 − 15 − 2022 − 283

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

flows. These methods are implemented in various computing packages. It should be noted that despite the convergence and stability of the solution when using the Kurganov-Tadmor scheme, the approach is dissipative, as a result of which it is necessary to use a finer grid, which leads to high costs for RAM and causes difficulties with a large amount of stored data. Therefore, there is a need to optimize the grid and adapt it during the calculation; within the framework of this approach, it is possible to single out the open library AMReX [10, 11]. However, it is important to consider the accuracy and computational costs of each method before deciding which one to use.

Currently, reducing the noise levels from supersonic jets is a major concern in industries such as combustion chamber design, jet engines, and pollution control. There are several sources of noise in supersonic jets [12–16], including large-scale turbulence, small-scale turbulence, broadband noise from the interaction of shockwaves and hydrodynamic instabilities (Mach waves), and narrowband noise from resonant flow regimes between shockwaves and hydrodynamic instabilities (Screech tone). Understanding the interaction between the high-speed flow, instabilities, and the environment is crucial in studying the acoustic noise of trans- and supersonic jets. To accurately predict noise in the far field, integral analogies for solving the Ffowcs-Williams and Hawkings equations can be used [20, 21]. In this work, this method was first implemented in the AMReX package for predicting the noise level, the results of this calculation were compared with the results for the libAcoustics library implemented in OpenFOAM.

In conclusion, the study of turbulent free jets has been an important area of research for many years due to its prevalence in nature and technology. The goal of reducing the noise level from supersonic jets remains an important area of study and research, with various approaches being taken to address this issue, including full-scale experiments, numerical simulations, and integral analogies. The choice of approach depends on a number of factors, including the accuracy of the method, computational costs, and the ability to study the acoustic noise level from the jet. Despite the progress made in this field, there is still much to be learned about turbulent free jets and their associated noise, and ongoing research continues to be conducted in this area.

2 Mathematical model and numerical method

For the calculation, a mathematical model is used, including an assembly for a compressible flow:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{j_m} = 0, \tag{1}$$

$$\frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{j_m} \otimes \mathbf{U}) + \nabla p = \nabla \cdot \hat{\sigma}, \tag{2}$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\mathbf{j_m} E) + \nabla \cdot \mathbf{q} = \nabla \cdot (\hat{\sigma} \mathbf{U}), \tag{3}$$

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

$$p = \rho R T, \tag{4}$$

where ρ - density; $\mathbf{j_m}$ - mass flux density; $\mathbf{U}$ - velocity vector; $E = e + |\mathbf{U}|/2$ - total energy, e - specific internal gas energy; $\hat{\sigma}$ - viscous stress tensor, $\mathbf{q}$ - heat flux. For all solvers, the mathematical model described above is used, but the numerical algorithm for solving them is different and has some peculiarities:

for pimpleCentralFoam [8, 22] and AMReX CNS solver [10, 11]:

$$\mathbf{j_m} = \rho \mathbf{U},\ \hat{\sigma} = \hat{\sigma}_{NS} = \mu[(\nabla \otimes \mathbf{U}) + (\nabla \otimes \mathbf{U})^T]; \tag{5}$$

for QGDSolver [9, 23, 24]:

$$\mathbf{j_m} = \rho(\mathbf{U} - \mathbf{w}),\ \hat{\sigma} = \hat{\sigma}_{NS} + \hat{\sigma}_{QGD},\ \mathbf{q} = \mathbf{q}_{NS} + \mathbf{q}_{QGD}. \tag{6}$$

2.1 OpenFOAM software, hybridCentralSolvers

The pimpleCentralFoam solver [22] is used for numerical simulation. This solver uses the operator splitting technique for the system of partial differential equations describing the low-speed motion of the fluid. For high-speed flows, the explicit Godunov-type methods are used. Two approaches were merged in the single hybrid method, proposed and developed by Kraposhin, for the simulation of flows in a wide range of Mach numbers [8]. Within this approach, the standard techniques for temporal derivatives, diffusion, and source terms are mixed with the KT/KNP fluxes for the convective terms. The KT/KNP convective fluxes are formulated for the unknown fields from the new time layer, yielding to the implicit approximation of a convection-diffusion equation. The modified PIMPLE algorithm is employed to couple pressure, velocity, and density. More details about the code, including the governing equations, can be found in the paper [8].

2.2 OpenFOAM software, QGDSolver

The QGDFoam solver [23] is used in the study, in which a numerical algorithm for solving regulated quasi-gasdynamic (QGD) equations is implemented. Being an extension of the classical system of Navier-Stokes equations, QGD systems contain additional terms that are proportional to the small coefficient τ, which has the dimension of time [9]. When the parameter τ tends to zero, the QGD system of equations transitions to the system of Navier-Stokes equations. In dimensionless form, the value of τ is proportional to the Knudsen number. For density gases, the value of τ is too small to use its direct value, since it does not provide the required stability of the numerical algorithm. In this case, the role of the free path in the numerical algorithm can be played by the computational grid step in space:

$$\tau = \alpha_{QGD} \frac{\Delta_h}{a},$$

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

where $\alpha_{QGD} \in [\,0, 1]$ is a constant, which is the tuning parameter of the numerical QGD algorithm, Δ_h is the size of the calculation cell, a is the speed of sound of a mixture of gases. When solving problems with high numbers Ma and Re, the introduced dissipation with the help of τ-terms is not enough, and therefore an additional viscosity is introduced into the system in the form of a coefficient in the viscous stress tensor σ: $\mu \to \mu + p\tau Sc_{QGD}$, where Sc_{QGD} is a scheme parameter that ensures its stability at high values of the local Ma number. As mentioned earlier, the variables w, σ_{QGD}, q_{QGD} - quasi-gasdynamic parameters that depend on τ. More details about the code, including the governing equations, can be found in paper [9, 24].

2.3 AMReX software, CNS solver

AMReX [10] is a C++ framework that supports the development of block-structured adaptive mesh refinement algorithms for solving partial differential equations (PDE) systems with complex boundary conditions for current and new numerical method architectures. The flow solver is implemented within the block-structured adaptive mesh refinement (AMR) framework of AMReX [18]. To solve the Navier-Stokes equations, finite-volume schemes of the second order of accuracy are used. The second-order Runge-Kutta method is used for the temporal discretization. This software package allows use block-orthogonal grids and automatic grid adaptation according to the selected parameter, while the equations on the new adaptation layer follow internal time. To solve the system of conservation equations, the third-order least squares method is used to calculate the velocity gradients and the CNS solver. The libAcoustics library is used together with AMReX CNS solver for sound pressure prediction of the jet, the calculation results are validated on experimental data and are presented in Section 4.

2.4 libAcoustics library

The Farassat $1A$ [20] formulation implemented in the libAcoustics library developed by the authors [25, 26] is used. This analogy is used to define the far field noise generated by an acoustic source moving through a gas. This library was verified on the problems of calculating noise from a monopoly and dipole source. And validation was carried out on a number of inkjet tasks, the results are published in papers [27, 28]. In this study, this library has been adapted for use in conjunction with the AMReX package. The following formula is used to calculate the sound pressure level (SPL):

$$SPL\,(dB) = 20log_{10}\left(\frac{p_{rms}}{p_{ref}}\right),$$

where p_{rms} is the RMS sound pressure, p_{ref} is the reference sound pressure. The developed library based on the OpenFOAM package is in the public domain [25] and can be compiled independently of any modules of the main package and the type of solvers.

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

3 Computational Setup

The problem of the high-speed free air jets flow from a round tube with a nozzle exit diameter $D = 0.01m$ into a space flooded with air is considered. The initial data correspond to the values of the parameters $M = U_j/a = U_j/\sqrt{kRT_j} = 2.1$, $Re = \rho_j U_j D/\mu_j = 70000$, where $U_j = 526m/s$ - jet exit velocity, $T_j = 156K$ - jet exit temperature, ρ_j - jet exit density, $k = 1.4$ - isentropic expansion factor, $p_j = 5066Pa$ - jet exit pressure, $p_c = p_j$ - chamber pressure, $p_a = 101325Pa$ - ambient pressure, $T_0 = 294K$ - constant stagnation temperature. The flow parameters and geometry correspond to the experimental study carried out by Troutt [3]. The initial condition for the velocity at the inlet is set given by the equation:

$$U(r) = 0.5 \cdot U_j \cdot \left[1 + tanh\left(10 \cdot \left(1 - \frac{2r}{D}\right)\right)\right].$$

The computational domain is a rectangular parallelepiped, in which the outlet boundary is removed by $100D$, and the side boundaries are removed by $20D$. The inlet boundary corresponded to the round nozzle exit and coincides with the origin of coordinates (see Fig. 1).

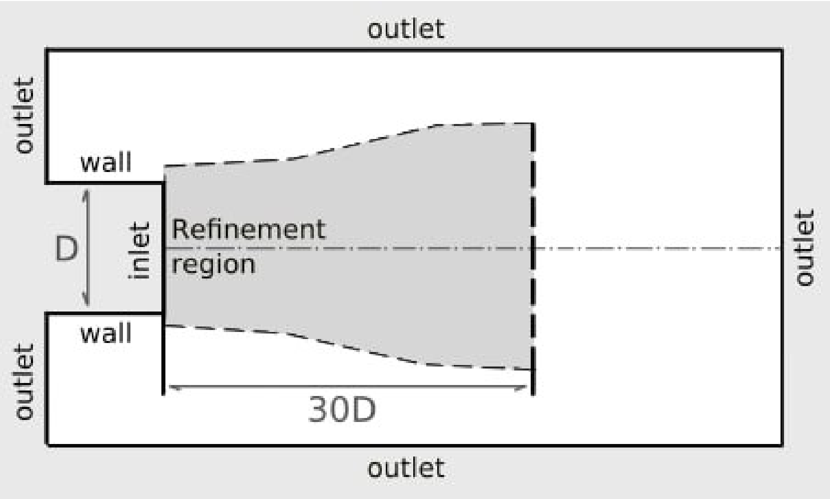
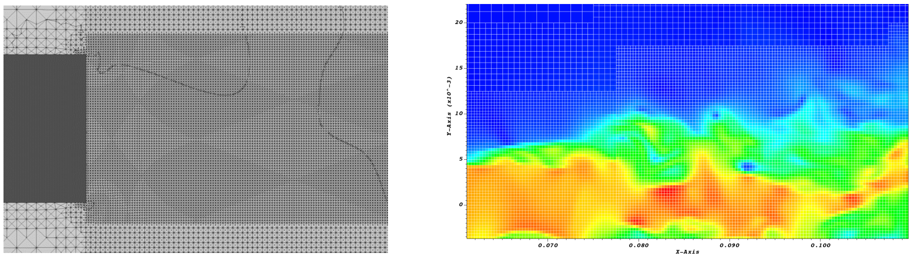


Fig. 1. Computational domain geometry.

To use solvers based on OpenFOAM, the computational grid is additionally refined by $30D$ downstream. Based on the recommendations presented in [19, 27], in this area of refinement, a computational grid with a resolution of 32 cells per diameter (CPD) is used. However, during the evaluation calculations, it is found that at a moderate Reynolds number and such a grid resolution, the hybrid solver poorly reproduces hydrodynamic instabilities. As a result, for hybridCentralSolvers, a computational grid is made with additional local refinement along

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

the jet axis with a resolution of $60CPD$ and a total number of cells of the order of 38 million (see Fig. 2). In the AMReX package, adaptive mesh refinement is performed according to the local Reynolds number. The mesh resolution in the region of the jet core is $32CPD$. The maximum number of cells in the calculation process is about 45 million.



a) b)

Fig. 2. A fragment of the computational grid in: (a) OpenFOAM; (b) AMReX

The virtual microphones are located at a distance $R = 40D$, the microphone position angle is set from 15 to 90 degrees. The pimpleCentarlFoam solver used the vanLeer scheme. In the QGDFoam solver, the following tuning parameters are used, which are defined in [27]: $\alpha_{QGD} = 0.15$, $Sc_{QGD} = 0$. For CNS, the criterion for mesh refinement by the local Reynolds number.

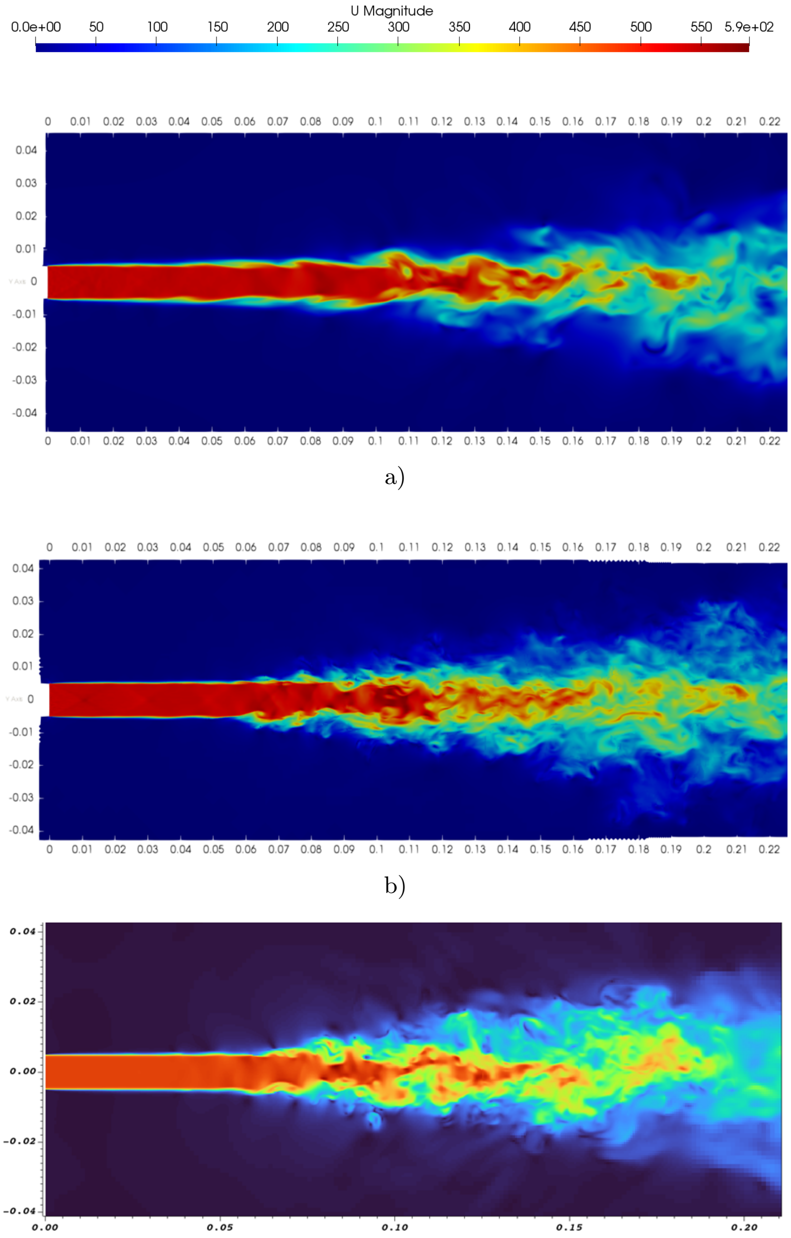
4 Results and discussion

Figure 3 shows that the hybrid pimpleCentralFoam solver is more dissipative, the QGD algorithm and the CNS solver more correctly reproduce the process of formation and propagation of hydrodynamic instabilities. So it makes sense to compare the ability of these two solvers to describe both pressure waves and temperature distribution in the jet (Fig. 4).

According to the figure 4a for QGDFoam solver and figure 4b for AMReX CNS solver for dimensionless pressure and temperature the CNS solver better describes the propagation of pressure waves in space due to adaptive mesh refinement. While in the QGDFoam solver, pressure waves are immediately attenuated in the coarse mesh region when propagating beyond the open control surfaces surrounding the jet flow. Based on the recommendations [29, 30] on choosing the shape of the control surface for calculating the acoustic pressure from jet flows, an open surface is constructed, which is schematically shown in Fig. 4. Figure 5 shows the axial distribution of the time-averaged Mach number and comparison of noise with experimental data.

The figures 3 and 5a show that due to the use of a structured grid and a higher order solver based on the AMReX package, it is possible to obtain a good match with the QGD algorithm which used mesh with more refinement. In this

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24



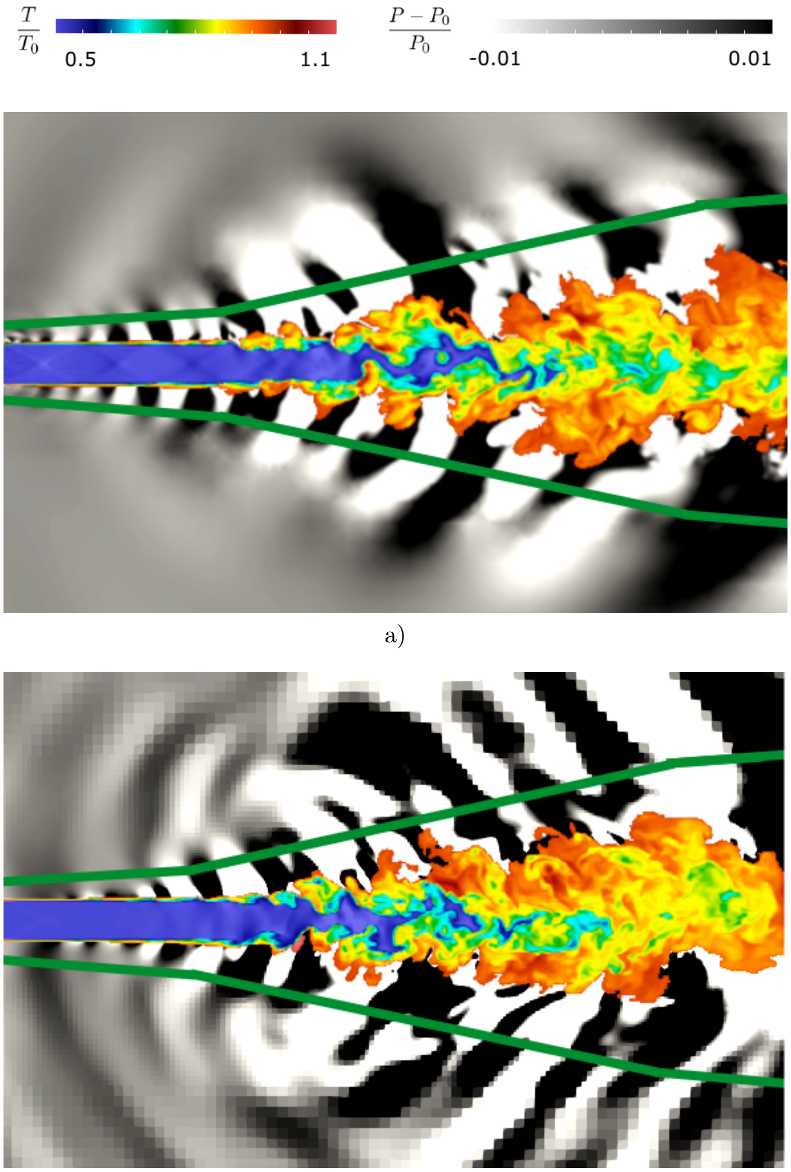
a)

b)

c)

Fig. 3. Instantaneous jet velocity distribution at M=2.1, Re=70000: (a) pimpleCentralFoamSolver; (b) QGDFoam solver (c) CNS solver

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

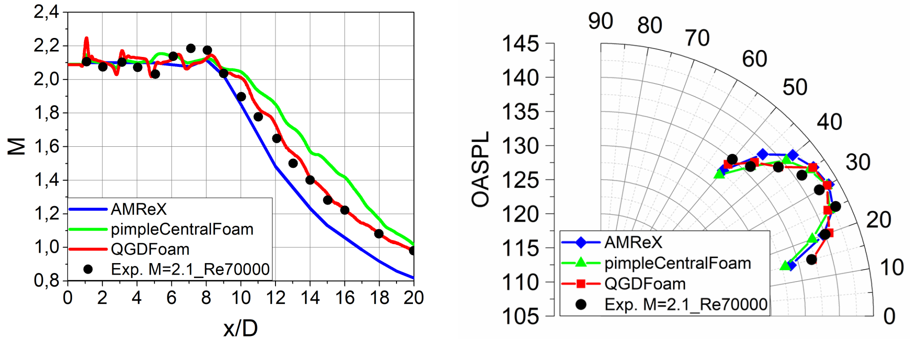


a)

b)

Fig. 4. Dimensionless fields of pressure and temperature at $M = 2.1$, $Re = 70000$, $T_0 = 156K$, $P_0 = 5066Pa$; green line - schematic showing the open control surfaces surrounding the jet flow: (a) QGDFoam solver (b) AMReX CNS solver

case, all three solvers give good agreement with the acoustic characteristics of the jet. The highest levels of generated noise occur at angles around 30^o.



a) b)

Fig. 5. Flow characteristics: (a) Axial distribution of the mean Mach number; (b) Sound pressure level directivity distributions

5 Conclusion

In modern research, numerical simulation of physical problems plays an important role, as a result of which the choice of an appropriate algorithm for numerical simulation is of paramount importance. Numerical studies of the applicability of various numerical algorithms for calculating supersonic flows implemented in various open-source solvers such as pimpleCentralFoam, QGDFoam, AMReX CNS solver are carried out. Cross-validation is performed on the example of calculating the ideally-expanded viscous gas jet and the acoustic noise generated by it at a $M = 2.1$ and a $Re = 70000$. For the first time, the libAcoustics library is used together with AMReX software for sound pressure propagation and further comparison with the results obtained in other solvers are conducted.

The space-time fields of gas-dynamic parameters of the near field and the acoustic pressure in the far field are defined. The analysis of the obtained data made it possible to determine the settings of numerical algorithms for solving such a class of the problems. All calculation results are obtained on the same grid resolution greater than $32CPD$. The results for the hybrid approach match the experimental data worse than the QGD algorithm and the solver based on block adaptive technology. The hybrid solver is more dissipative than the other considered algorithms and requires a more detailed grid in the region of the jet core in order to correctly reproduce hydrodynamic instabilities. The QGD algorithm, with regularization parameters $Sc = 0$, $\alpha_{QGD} = 0.15$ makes it possible to accurately describe the flow structure and acoustic characteristics, but due to the implementation features, it requires more computational time. An approach based on a block-structured adaptive grid makes it possible to obtain a more accurate result with the same grid resolution in the region of the jet core. This is due to the higher order of approximation scheme that can be used on the structured grids. Therefore, the CNS solver would be reasonable to use in such cases where it is necessary to calculate the propagation of turbulent jets over long distances.

Acknowledgements This work was supported by Moscow Center of Fundamental and Applied Mathematics, Agreement with the Ministry of Science and Higher Education of the Russian Federation, grant number $075-15-2022-283$.

References

1. Ginevsky, AS Theory of turbulent jets and traces: Integral methods of calculation. Engineering, (1969).
2. Stromberg, J.L., McLaughlin, D.K., Troutt, T.R.: Flow field and acoustic properties of a Mach number 0.9 jet at a low Reynolds number. J. Sound Vibr. 72(2), 159–176 (1980)
3. Troutt, T.R., McLaughlin, D.K.: Experiments on the flow and acoustic properties of a moderate-Reynolds-number supersonic jet. J. Fluid Mech. 116, 123–156 (1982)
4. Biswas S., Qiao L. A numerical investigation of ignition of ultra-lean premixed H/air mixtures by pre-chamber supersonic hot jet //SAE International Journal of Engines. – 2017. – . 10. – 5. – . 2231-2247.
5. Li X. R. et al. Acoustic feedback loops for screech tones of underexpanded free round jets at different modes //Journal of Fluid Mechanics. – 2020. – . 902. – . A17.
6. B.G. Galerkin, "On electrical circuits for the approximate solution of the Laplace equation" Vestnik Inzh. , 19 (1915) pp. 897–908 (In Russian)
7. Epikhin, A., Kraposhin, M., Vatutin, K.: The numerical simulation of compressible jet at low Reynolds number using OpenFOAM. In: E3S Web of Conferences, vol. 128 (2019)
8. Kraposhin, M., V., Banholzer, M., Pfitzner, M., Marchevsky, I., K.: A hybrid pressure-based solver for non ideal single-phase fluid flows at all speeds. International Journal for Numerical Methods in Fluids 88(2), 79-99 (2018).
9. Elizarova, T.,G.: Quasi-gas dynamic equations, Springer, Berlin (2009).
10. AMReX Guided Tutorials, https://amrex-codes.github.io/amrex/tutorials Last accessed 3 Feb 2023
11. AMReX CNS flow solver, https://github.com/AMReX-Codes/amrex/tree/development/Tests/EB/CNS Last accessed 3 Feb 2023
12. Baars, W.J., Tinney, C.E., Murray, N.E., Jansen, B.J., Panickar, P.: The effect of heat on turbulent mixing noise in supersonic jets. AIAA Paper, 2011-1029 (2011)
13. Tam, C.K.W., Viswanathan, K., Ahuja, K.K., Panda, J.: The sources of jet noise: experimental evidence. J. Fluid Mech. 615, 253–292 (2008)
14. Tam, C.K.W., Shen, H., Raman, G.: Screech tones of supersonic jets from bevelled rectangular nozzles. AIAA J. 35(7), 1119–1125 (1997)
15. Tam, C.K.W., Burton, D.E.: Sound generated by instability waves of supersonic flows. Part 2. Axisymmetric jets. J. Fluid Mech. 138, 273–295 (1984)
16. Tam, C.K.W.: Mach wave radiation from high-speed jets. AIAA J. 47(10), 2440–2448 (1984)
17. D. N. Arnold, F. Brezzi, B. Cockburn and L. Marini, Unified analysis of discontinuous Galerkin methods for elliptic problems, SIAM J. Numer. Anal. 39 (2002), no. 5, 1749–1779.

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24

18. Natarajan M. et al. A moving embedded boundary approach for the compressible Navier-Stokes equations in a block-structured adaptive refinement framework //Journal of Computational Physics. (2022)
19. Epikhin, A., Kraposhin, M., Vatutin, K.: The numerical simulation of compressible jet at low Reynolds number using OpenFOAM. In: E3S Web of Conferences, vol. 128 (2019)
20. Brentner, K., S., Farassat, F.: An analytical comparison of the acoustic analogy and Kirchhoff formulations formoving surfaces. AIAA Journal 36, 1379-1386 (1998).
21. Brès G., A., Pérot, F., Freed, D.: A Ffowcs Williams–Hawkings solver for lattice Boltzmann based computational aeroacoustics. AIAA Paper, 2010-3711, (2010).
22. hybridCentralSolvers https://github.com/unicfdlab/hybridCentralSolvers Last accessed 3 Feb 2023
23. QGDSolvers, https://github.com/unicfdlab/QGDsolver Last accessed 3 Feb 2023
24. Kraposhin, M., V., Smirnova, E., V., Elizarova, T., G., Istomina, M., A.: Development of a new OpenFOAM solver using regularized gas dynamic equations. Computers and Fluids 166, 163-175 (2018).
25. libAcoustics library, https://github.com/unicfdlab/libAcoustics Last accessed 3 Feb 2023
26. Epikhin, A., Evdokimov, I., Kraposhin, M., Kalugin, M., Strijhak, S.: Development of a dynamic library for computational aeroacoustics applications using the OpenFOAM open source package. Procedia Computer Science 66, 150-157 (2015).
27. Epikhin, A., Kraposhin, M. (2020). Prediction of the Free Jet Noise Using Quasi-gas Dynamic Equations and Acoustic Analogy. Lecture Notes in Computer Science, vol 12143.
28. Melnikova, V.G.; Epikhin, A.S.; Kraposhin, M.V. The Eulerian–Lagrangian Approach for the Numerical Investigation of an Acoustic Field Generated by a High-Speed Gas-Droplet Flow. Fluids (2021)
29. Uzun, A., Lyrintzis, A., S., Blaisdell, G., A., 2004, Coupling of integral acoustics methods with LES for jet noise prediction, AIAA Paper, pp. 4982-5001.
30. Shur, M., and Spalart, P., Strelets, M., 2005, Noise prediction for increasingly complex jets. Part I: Methods and tests, International Journal of Aeroacoustics, 4, pp.213-246.

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_24